

Test Problems for IWPCTM 11

Test Problem 2: Multi-mode implosion in cylindrical 3D geometry

At the start of the problem, a cylinder of light fluid 1 is surrounded by a shell of dense fluid 2. A multimode perturbation is present at the interface. The implosion is driven by pressure on the outside of the dense shell. The problem may be formulated in one of two ways (a) use a Lagrangian mesh boundary at the outside of fluid 2 with the required applied pressure as a function time or (b) add a extra region outside the dense shell in which the pressure is specified (as assumed below). If the problem is run using a cylindrical polar mesh, an angular extent of $\pi/4$ is recommended. For a Cartesian mesh one quadrant, $\pi/2$, should be used. The initial radius of the light cylinder is unity (any consistent set of units may be used).

Initial densities ρ , outer radii R_o , and pressures P are:

Region 1: fluid 1, $\rho = 0.05$ g/cc, $R_o = 1$, $P = 0.1$

Region 2: fluid 2, $\rho = 1.00$ g/cc, $R_o = 1.2$, $P = 0.1$

Region 3: fluid 1, $\rho = 0.05$ g/cc, $R_o = 1.5$,

$P = 13$ for $t \leq 0.04$

then linear ramp to $P=0.5$ at $t = 0.125$

$P = 0.5$ for $0.125 \leq t \leq 0.3$

The length of the cylinder, in the z-direction, is $L=0.25$.

A perfect gas law is used; $\gamma=5/3$ for both fluids. The ratio of specific heats is $C_{v1}/C_{v2} = 20$ – this gives uniform initial temperature. (This ratio is only needed to relate volume fractions to mass fractions).

Random initial perturbations are applied to the outside of the light cylinder – i.e. the initial radius is perturbed from $r = R_I$ to $r = R_I + \zeta(\theta, z)$. A typical ICF surface spectrum is used for ζ :-

$$\langle \zeta^2 \rangle \equiv \int dk P(k) \quad \text{where} \quad P(k) \sim \frac{1}{k^2} \quad \text{for } 0.01 < \lambda < 0.2 \quad \text{and} \quad \lambda = 2\pi / k$$

$$\sqrt{\langle \zeta^2 \rangle} = 0.00005$$

FORTTRAN code to calculate the perturbation is provided. This calculates ζ as a function of distance coordinates $s = R_1\theta$ and z . The user should input appropriate equally spaced s and z values (s_j, z_k). The output array ζ_{jk} may then be used to perturb the surface i.e set the initial volume fractions for fluids 1 and 2. Reflective or periodic boundary conditions can be used when calculating the perturbation. The recommendation is reflective in the θ -direction, periodic in the z -direction. The perturbation has the following form:-

$$\zeta(s, z) = S \sum_{m,n} a_{mn} \cos\{mk_0(s - s_1)\} \cos\{n\ell_0(z - z_1)\} + b_{mn} \cos\{mk_0(s - s_1)\} \sin\{n\ell_0(z - z_1)\} + c_{mn} \sin\{mk_0(s - s_1)\} \cos\{n\ell_0(z - z_1)\} + d_{mn} \sin\{mk_0(s - s_1)\} \sin\{n\ell_0(z - z_1)\}$$

The coefficients $a_{mn}, b_{mn}, c_{mn}, d_{mn}$ are chosen from Gaussian distributions with

$$s.d \sim \sqrt{P(k)/k}, \quad k = \sqrt{(mk_0)^2 + (n\ell_0)^2}$$

The minimum wavenumbers are

$$k_0 = \begin{cases} 2\pi / W_\theta & \text{periodic bouyndary conditions} \\ \pi / W_\theta & \text{reflective bouyndary conditions} \end{cases} \quad \ell_0 = \begin{cases} 2\pi / W_z & \text{periodic bouyndary conditions} \\ \pi / W_z & \text{reflective boundary conditions} \end{cases}$$

The scale factor S is chosen to give the required s.d.

TURMOIL uses a cylindrical polar mesh, Lagrangian in the r -direction. For the “standard” mesh, the initial radial zoning is :-

0.0 < r < 0.25	75 meshes: 1D Lagrangian zoning
0.25 < r < 1.0	225 meshes: 3D region
1.0 < r < 1.13	130 meshes: 3D region
1.13 < r < 1.2	70 meshes: 1D Lagrangian zoning

256 meshes are used in the θ - direction (0 to $\pi/4$) with reflective boundary conditions. 256 meshes are used in the z -direction (0 to 0.25) with periodic boundary conditions. It is recommended that mesh resolution comparable to this, or finer, should be used for all simulations.

The calculation should be run to $t=0.3$ (maximum compression occurs at about $t=0.24$) and the output described below should be obtained.

If sub-grid pressure and temperature equilibrium is assumed, then fluid volume fractions, f_s , may then be expressed in terms of mass fractions, m_s :

$$f_s = \frac{(\gamma_s - 1)c_{vs}m_s}{(\gamma_1 - 1)c_{v1}m_1 + (\gamma_2 - 1)c_{v2}m_2}$$

The following quantities should then be plotted:-

(a) The unperturbed interface radius, R_I , versus time

(b) Mix widths, $R_I - X_1$ (spike) and $X_2 - R_I$ (bubble), versus time.
 X_1 and X_2 are the values of radius for which $\bar{f}_1 = 0.01$ and $\bar{f}_2 = 0.01$.
 where \bar{C} denotes the value of C averaged over θ and z .

(c) Integral mix width, W , versus time.

This is defined as

$$W = \int \bar{f}_1 \bar{f}_2 dr$$

(d) Mean fluid 2 volume fraction, \bar{f}_2 , versus radius at $t=3$.

(e) Resolved turbulence kinetic energy, k , versus radius at $t=3$

This is defined as

$$k = \frac{1}{2} \bar{n} \{ (u_r - \tilde{u}_r)^2 + u_\theta^2 + u_z^2 \} / \bar{n}$$

where \tilde{u}_r is the mass- weighted mean radial velocity

(f) Molecular mixing fraction, Θ , versus radius at $t=3$.

This is defined as:-

$$\Theta = \frac{\overline{f_1 f_2}}{\bar{f}_1 \bar{f}_2}$$

One-dimensional engineering model calculations can also be run on this test problem. The quantities plotted should be the same as for 3D simulations.

The figure below gives a plot of R_I , X_1 and X_2 versus time for the standard mesh TURMOIL simulation. The data (x,y pairs) is supplied in a text file.

